SC16 Content Submission

Instructions

Thank you for participating in SC16. The information you provide here will be used to create conference posters, web content, press kit material, and other NASA media products to promote your work. If your demo covers research presented at previous SC conferences, please provide new and updated text and images to reflect progress over the last year. If you have any questions, please send email to: scupload@nas.nasa.gov

Creating and Submitting Your Content

1. Before you add any content, please save and rename this file by using the **Save As** feature in Word. Use the following naming convention: *LastName_FirstInitial_DemoKeyword.doc*

2. Enter your abstract content directly into the text box at far right, noting the important instructions, guidelines, and tips. Leave the subheads intact, and replace the text in the box with your info. Keep the content as public-friendly as possible. Refer to the example abstract in the middle column for content length and tone. You can also see more example abstracts on the SC16 website.

3. Enter your poster content by going to page 2 of this document. Please enter your information directly into the light gray text boxes provided, replacing the text with your own. The background is a simplified version of this year's poster design. Your poster will have different colors and design details depending on your theme and mission directorate. See page 3 for an example of a completed poster from a past SC conference.

4. Format your two images: All images must have a resolution of at least 2,000 x 2,400 pixels at full size, or 14" x 10" at 200 dpi. Please submit only the following standard formats: .eps, .tif, .png, .psd, .jpg

5. Upload your completed form, poster images, and any optional video files no later than Friday, September 9, 2016 to: https://www.nas.nasa.gov/cgi-bin/upload/sc16upload

   Use your NAS Lou username and password to log in.

Videos and Extra Image - Optional

If you have 1-2 video clips/movies (and/or a third image) relevant to your NASA demo at SC16, please submit these assets as described below. Movies and images may be used in the NASA booth, on printed materials, or on the NASA SC16 website.

1. Add those files to the upload site.
2. Submit video/extra image captions and credits via email to: scupload@nas.nasa.gov with the following information:

SUBJECT: SC16 Video Captions for LastName_FirstInitial DemoKeyword

Filename: *LastName_FirstInitial_DemoKeyword.mpg*
Caption: 60 words maximum
Video Credit: FirstName LastName, NASA/Center (2 names maximum)

Video Requirements:
Runtime: 30 seconds maximum
Frame size: 720x480 minimum, 1920x1080 maximum
Frame rate: 10fps minimum, 30fps maximum
Preferred formats: .mov, .mp4 Acceptable formats: .mpg, avid, .wmv

Sample Abstract from Past Conference

Innovating the Future of the Passenger Airplane

Overview
NASA is investigating innovative new ways of improving passenger airplanes. Higher fuel efficiency and less noise are two major goals of this effort. One proposed idea—the D8 airplane—takes its inspiration from the submarine. The D8 design uses a doublewide fuselage configuration made by joining two traditional fuselages placed side-by-side. The wider fuselage provides higher lift and two aisles for boarding passengers. This reduces the amount of lift required from the wing, making it possible to design a smaller, lighter wing. The D8's engines are mounted on the top rear of the fuselage, which enables the fuselage and tail assembly to act as a diffuser and nacelle, eliminating the need for an additional nacelle to house the engine components. The design also includes a boundary-layer ingesting fan that should provide additional efficiency as it does for a submarine propeller.

Project Details
A 1:11 scale model of the D8 is being tested in the 14-by-22-foot wind tunnel at Langley Research Center to ascertain whether the concept actually uses less power to fly compared to a conventional design. In conjunction with these tests, computational fluid dynamics (CFD) simulations are being performed to determine the effect of the boundary-layer ingesting fan on the airplane's performance. CFD simulations of both the full-scale airplane and the wind tunnel model are being conducted. An unpowered model of the airplane is used to validate the simulations. Subsequently, the power required to fly the aircraft is computed for a podded nacelle configuration, which has the engines placed on pods on the side of the fuselage like a business jet, and the boundary-layer ingesting configuration with the engines placed on the top rear of the fuselage. The required power is then compared to determine which method is more efficient.

Results and Impact
The computational results are expected to verify the wind tunnel experiment's conclusion that the D8's integrated nacelle design is more fuel-efficient than the traditional podded nacelles. If proven, this could mean that future aircraft may use this NASA-fostered technology to save fuel on commercial aircraft. This fuel savings may translate to lower air travel costs for passengers. The computational results are also expected to shed additional light on the reasons behind the better fuel efficiency by examining the airflow behavior in the vicinity of the engines.

Role of High-End Computing Resources
CFD simulations of the aircraft in flight require accurate modeling of both the aircraft and the airspace in its vicinity. To accomplish this, a computational model of the full aircraft would require approximately 250 million grid cells. Computing a single analysis run for a model of this resolution would take approximately 30,000 processor-hours on the Pleiades supercomputer at NASA Ames Research Center. Our simulations use an assumption of symmetry, which makes it possible to compute flow on only one side of the airplane and reduces the computing requirements by half. The results allow us to compare computational data with wind tunnel tests to validate the solution accuracy and determine if the boundary-layer ingesting nacelle is indeed more fuel efficient. The fast NASA networks also allow a geographically separated team from two NASA centers and the Massachusetts Institute of Technology (MIT) to collaborate efficiently.

Michael Barad, NASA Ames Research Center
michael.f.barad@nasa.gov

Shishir Pandya, NASA Ames Research Center
shishir.pandya@nasa.gov

Aeronautics

See more example abstracts on the SC14 website.

Enter Abstract Below (*double-click in box to edit*)

Improving Fidelity of Launch Vehicle Liftoff Acoustic Simulations

Overview
Launch vehicles experience high acoustic loads during ignition and liftoff affected by the interaction of rocket plume generated acoustic waves with launch pad structures. Application of highly parallelized Computational Fluid Dynamics (CFD) analysis tools optimized for application on the NAS computer systems such as the Loci/CHEM program now enable simulation of time-accurate, turbulent, multi-species plume formation and interaction with launch pad geometry and capture the generation of acoustic noise at the source regions in the plume shear layers and impingement regions. These CFD solvers are robust in capturing the acoustic fluctuations, but they are too dissipative to accurately resolve the propagation of the acoustic waves throughout the launch environment domain along the vehicle.

A hybrid Computational Fluid Dynamics and Computational Aero-Acoustics (CFD/CAA) modeling framework has been developed to improve such liftoff acoustic environment predictions. The framework combines the existing highly-scalable NASA production CFD code, Loci/CHEM, with a high-order accurate discontinuous Galerkin (DG) solver, Loci/THRUST, developed in the same computational framework. Loci/THRUST employs a low dissipation, high-order, unstructured DG method to accurately propagate acoustic waves away from the source regions across large distances. The DG solver is currently capable of solving up to 4th order solutions for non-linear, conservative acoustic field propagation. Higher order boundary conditions are implemented to accurately model the reflection and refraction of acoustic waves on launch pad components. The DG solver accepts generalized unstructured meshes, enabling efficient application of common mesh generation tools for CHEM and THRUST simulations. The DG solution is coupled with the CFD solution at interface boundaries placed near the CFD acoustic source regions. Both simulations are executed simultaneously with coordinated boundary condition data exchange.

Project Details
Initial application testing and validation of the higher order acoustic propagation technique was carried out against acoustic data from the Ares Scale Model Acoustic Test (ASMAT) experiments. This simulation served to evaluate the capabilities and production readiness of the CFD/CAA framework towards resolving the experimentally observed spectrum of acoustic frequency content. Numerous improvements to the numerical algorithm implementation, boundary conditions and communication process between the simulations were identified and implemented during this project. The initial applications presented here were performed with the 3rd order DG solver. Testing with the 4th order solver will commence as soon as higher order boundary conditions have been fully implemented.

Results and Impact
The new CFD/CAA approach proved highly capable of accurately propagating and conserving the acoustic wave field over the complex launch vehicle and launch pad geometry Acoustic wave content was preserved to a significantly higher frequency range in the DG solver predictions, especially at sensor locations farther away from the plume source regions in the upper vehicle portions. This improved capability to perform high fidelity computational acoustic field simulations increases the confidence in the specification and understanding of launch acoustic loads design environments through computational modeling Understanding of the acoustic environments can be expanded beyond the data available from the limited number of sub-scale tests that could be performed for ASMAT and the current SMAT (SLS Model Acoustic Test). Numerical simulations can be applied in evaluating various sound suppression measures reducing the need for expensive testing.

Role of High-End Computing (Why HPC Matters)
NASA supercomputing resources are instrumental in completing this type of analysis. The computational model requires nearly 300 million mesh cells to simultaneously resolve the launch vehicle and launch pad details and adequately capture the acoustic sources at the rocket plumes. Both simulations are executed simultaneously exchanging transient boundary condition input to the DG solution, utilizing the sophisticated message passing enabled by the high performance NAS architectures. Each of the ASMAT simulations was performed using 2,000 parallel computing processors on NASA's Pleiades supercomputer. The dynamic simulation of the ignition dynamics and acoustic field propagation used nearly 2 million processor-hours over the time frame of two to three weeks, which would have been an intractable problem in the past.

Peter Liever, Jacobs ESSSA Group, NASA Marshall Space Flight Center
peter.a.liever@nasa.gov
Jeff West, NASA Marshall Space Flight Center
jeffrey.s.west@nasa.gov

Space Exploration

Please select one of the following themes and delete the others:

Space Exploration

Theme

Improving Fidelity of Launch Vehicle Liftoff Acoustic Simulations

Image

Image Placeholder #1
(Do not insert actual images here)

All images must have a resolution of 2,000 x 2,400 pixels at full size, or 14" x 10" at 200 dpi.

Please submit only the following standard formats: .eps, .tif, .png, .psd, .jpg

Image caption

The acoustic solver domain is embedded in the full CFD simulation domain. Acoustic wave input is received through boundary patches located near the acoustic source regions such as near the flame trench under the launch platform, the top of the flame trench and inside the launch mount.
Peter Liever, Jeff West, NASA/MSFC

Image: CFD-DG-Domains.png

Image caption

A comparison of the pressure wave field resolution for the CFD and the DG acoustic field solver demonstrates the increased resolution and preservation of acoustic waves propagating during the ignition process.
Peter Liever, Jeff West, NASA/MSFC

Image:
CFD-DG-PressureField.png

Image

Image Placeholder #2
(Do not insert actual images here)

All images must have a resolution of 2,000 x 2,400 pixels at full size, or 14" x 10" at 200 dpi.

Please submit only the following standard formats: .eps, .tif, .png, .psd, .jpg

Main contents

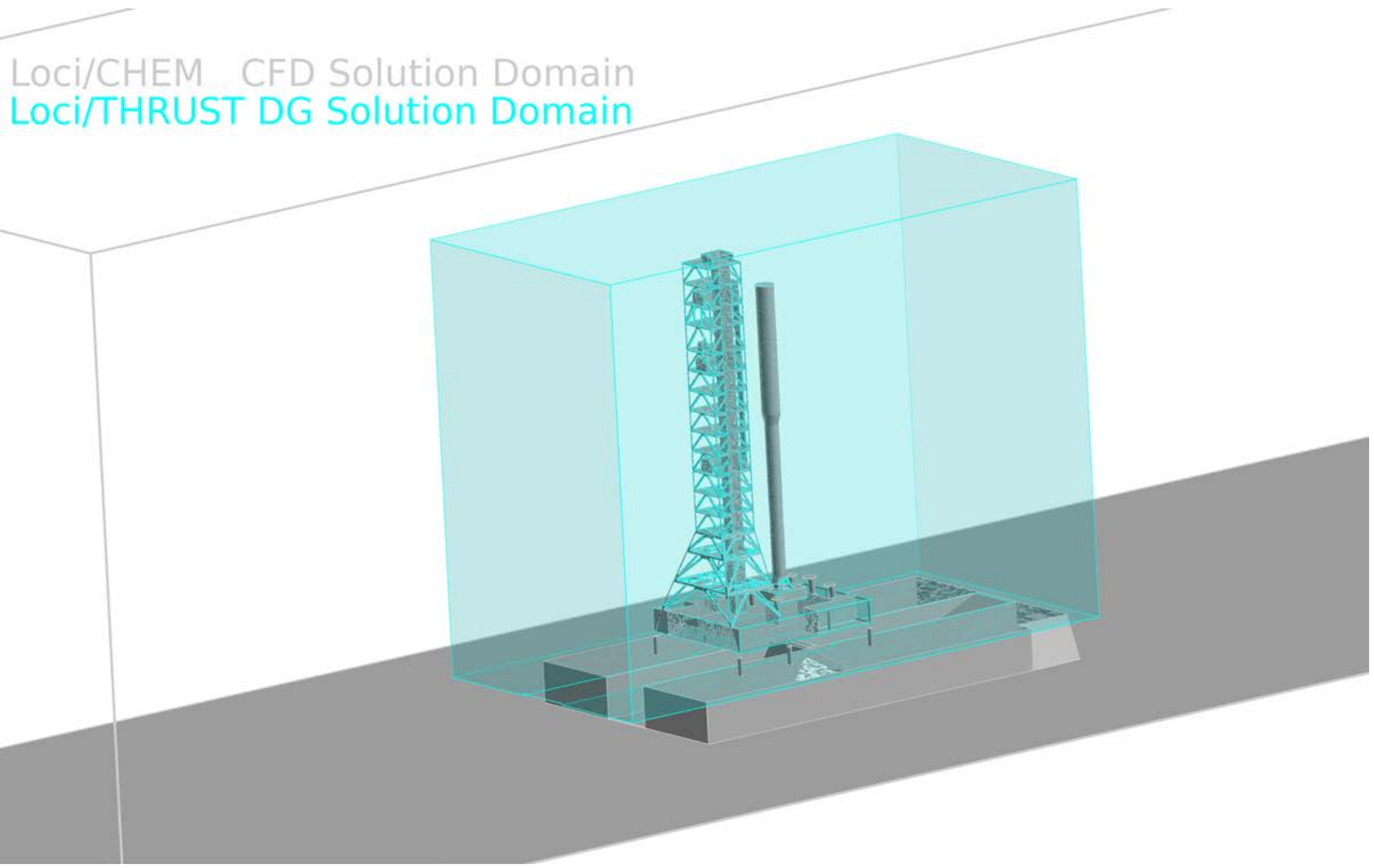
Launch vehicles experience high acoustic loads during ignition and liftoff from rocket plume generated acoustic waves with launch pad structures. Computational Fluid Dynamics (CFD) analysis tools optimized for application on the NAS computer systems now enable simulation of time-accurate, turbulent, multi-species plume formation and interaction with launch pad geometry and capture the generation of acoustic noise at the source regions in the plume shear layers and impingement regions. These CFD solvers are robust in capturing the acoustic fluctuations, but they are too dissipative to accurately resolve the propagation of the acoustic waves throughout the launch environment domain along the vehicle. A hybrid Computational Fluid Dynamics and Computational Aero-Acoustics (CFD/CAA) modeling framework has been developed to improve liftoff acoustic environment predictions.

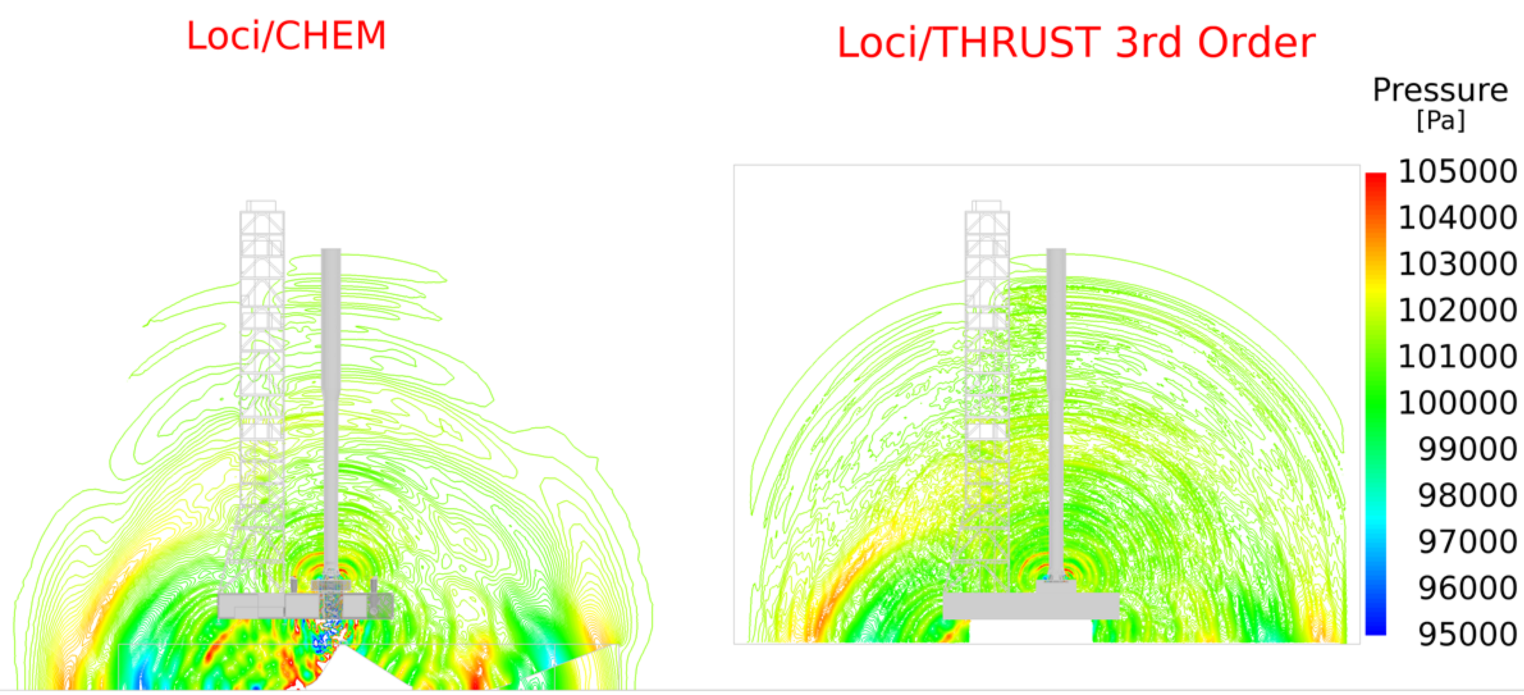
- The framework combines the existing highly-scalable NASA production CFD code, Loci/CHEM, with a high-order accurate discontinuous Galerkin (DG) solver, Loci/THRUST, developed in the same computational framework.
- Loci/THRUST employs a low dissipation, high-order, unstructured DG method to propagate acoustic waves from source regions across large distances and capturing the wave interaction with the complex launch pad structures.
- The DG solution is coupled with the CFD solution at interface boundaries placed near the CFD acoustic source regions.

Initial application testing and validation of the higher order acoustic propagation technique was carried out against the Ares Scale Model Acoustic Test (ASMAT) experiments. The new CFD/CAA approach proved highly capable of accurately propagating and conserving the acoustic wave field over the complex launch vehicle and launch pad geometry. This improved capability to perform high fidelity computational acoustic field simulations will increase the confidence in the characterization of launch acoustic loads environments through computational modeling

Peter Liever, Jacobs ESSSA Group, NASA Marshall Space Flight Center

Jeff West, NASA Marshall Space Flight Center


Loci/CHEM   CFD Solution Domain
Loci/THRUST DG Solution Domain


Loci/CHEM
Loci/THRUST 3rd Order
Pressure
[Pa]
105000
104000
103000
102000
101000
100000
99000
98000
97000
96000
95000